Premier Journal of Engineering and Applied Sciences
Publication of Nigerian Society of Engineers, Ibadan Branch
P J E A S
Vol. 1, No. 4, 2020 ISSN:2734-3170

Parametric Design of Shell and Tube Heat Exchangers Using Computational Fluid Dynamics (CFD): Different Materials Approach

A. E. Adeleke[a*], S. Oyelami[b], T. Ogundana[a] and O. Adetula[a]

[a]Department of Mechanical Engineering, Federal University Oye Ekiti, Nigeria
[b]Department of Mechanical Engineering, Osun State University, Osogbo, Nigeria

Corresponding Author: *ademola.adeleke@fuoye.edu.ng*

ARTICLE INFO

Received: August, 2020
Accepted: October, 2020
Published: October, 2020

Keywords:
ANSYS
Heat exchanger
Simulation
Heat transfer
Computational Fluid Dynamics (CFD)

ABSTRACT

The mechanical device that transfers energy from a hot fluid to a cold fluid is known as a heat exchanger. The production of heat exchanger of high thermal performance depends on successful parametric design. Modelling of the heat exchanger was carried out using the Solid works 2016 design module with the initial geometric dimensions taking as: number of tubes 12, number of baffles 6, length of tube 2004 mm, inner and outer diameters of tube are 65 mm and 70 mm respectively. Computer simulations of the thermo physical properties of different materials such as Aluminum, copper and steel were carried out to optimize the modelled heat exchanger using Computational Fluid Dynamics (CFD) technique. The wall heat transfer coefficient, velocity profile, pressure and temperature differences for the three materials were determined. A three-dimensional numerical simulation of fluid flow and heat transfer in a shell and tube heat exchanger has been carried out.

1. INTRODUCTION

Energy transmission due to temperature gradient from one region to another is termed heat transfer. There are three modes of heat transfer namely; conduction, convection and radiation. Many engineering applications involved exchange of heat between the fluids of different temperatures which are separated with solid wall (Rajput, 2012). Any mechanical system that involves transformation of working fluid from one phase to another require transfer of heat. Heat transfer has a significant role due to the demand of energy that keeps increasing and almost every aspect of an industry requires heat transfer. It is available in many different forms, particularly for the current use in the industrial sectors. Heat exchanger is used to achieve transfer of thermal energy during the following processes; power production, chemical processing, waste heat recovery, air conditioning and space heating (Kolepaka and Bicha, 2017).

The operation of heat exchanger does not involve interaction between external work and heat (Dawit, 2014; Mohammed and Sayed, 2020). In a shell and tube heat exchanger, fluids are separated by tube wall, one of the fluids flows through a bundle of tubes enclosed by a shell. The other fluid is forced through the shell and it flows over the outside surface of the tubes (Arjun and Gopu, 2014). To encourage highly efficient heat transfer process, selection of appropriate materials for the design of heat exchanger is required (Rajput, 2012; Nihat *et al.*, 2020). The study of science that deals with the prediction of the pattern of fluid flow, chemical reactions, mass and heat transfer and related properties by providing solution to set of mathematical governing equations like conversion of energy, mass, momentum, species, and effects of body forces is known as Computational Fluid Dynamics (CFD). It is a powerful tool for research and development in heat transfer simulation and modeling of heat exchanger in the multiphase flow systems.

There are several studies in the literature using fluid dynamics modeling for shell and tube heat exchanger, notably the studies by Yang *et al.* (1990); Diaper and Heseler (1990); Zhang and Sousa (1990); Haseler *et al.* (1992); Revesz (1992); Lorenzini *et al.* (1992); Prekdemir *et al.* (1993); Zhang *et al.* (1993); Keene *et al.* (1994); Prekdemir *et al.*(1994); Dilip *et al.* (2015); Huang *et al.*(1996); Santosh *et al.*(2017); Baru *et al.* (2018); Sneha *et al.* (2018) who, in most cases, used CFD to carry out the modelling and design of heat exchanger. Much are not been found in literature on the use of CFD and solid work in the design of shell and tube heat exchanger with varying different materials. This work optimize the effect of different materials on the design of the heat exchanger and predict the temperature, velocity distribution along the shell and tubes.

2. METHODOLOGY

Modelling and Simulation of the System

Heat exchanger was modeled using the Solid works 2016 design module. A counter-flow type of heat exchanger was modeled; it comprised of five parts which included shell, shell cover, tubes, baffle, and gaskets. The shell and tube heat exchanger (STHE) had two inlets (one on the shell and another on the head cover) and two outlets (one on the shell and another on the head cover). The head cover was designed to allow the fluid entering the tube through the inlet opening and fluid leaving the tube through the outlet opening of the head cover. The inlet and the outlet openings were placed side by side which was partitioned to allow efficient heat transfer after a long travelling of the fluid. The baffles were typically arranged with the tube which helps to hold the tube and directs the shell fluid flow to large extent.

The parameters used for the modelling are as shown in Table 1. The baffles were spaced in the correct orientation because this affects the pattern of flow of the shell side. There are twelve tubes inside the shell. The dimensions of the tube are; 90 mm inner diameter and 100 mm outer diameter and 2004 mm long which determines the overall heat transfer area. The pattern and the pitch (the center-to-center distance between the tubes) affects the pressure drop and dictates the number of heat tubes that can fit in the shell. The shell cover allows the fluid coming from the inlet tubes to travel into the outlet tubes. The model was imported and simulation was carried out on it using Ansys fluent solver to determine the variation of temperature and velocity along the tubes during the heat exchanging process. The following procedures were carried out using inbuilt parameters in the solver.

Table 1: Dimension of the Geometry and Sketch

Geometry name	Dimension
Tube length, Inner diameter and outer diameter	2004 mm,90 mm and 100 mm
Shell length, inner diameter and outer diameter	2004 mm, 550 mm, 650 mm
Numberof tubes	12
Baffle angle	90^0
Numbers of baffle, Nb	6
Baffle cut, bc	50 %
shell cover inner and outer diameter	630 mm, 650 mm
Number of gaskets	2

Definition of the Contacts and Selection of Names

In other to get the pattern of flow of fluid in the shell region, the baffles, tube assemblage and gasket were selected to be in contact with the shell. Likewise, the shell cover, head cover, tube assemblage were selected to be in contact with shell for the tube region. The face for the cold inlet was selected (bottom) and named *Inlet cold*, the selection of the hot inlet was done at the other side of the model (top) to avoid divergent problem during solution, and the face was selected and named *Inlet hot*. The cold outlet was selected for the corresponding region face of the cold at the other end and was named *outlet cold*. The same was done for hot outlet face *outlet hot.*

Meshing and Assumptions Made

In order to make the simulation more effective, meshing of small size was used. Meshing is the process of dividing the model into a number of elements which determine how uniformly distributed the load is when applied. The solid tubes were meshed using sweep mesh 0.005 meshing size. The mesh quality was determined using the statistical measure. The mesh metrics was generated to confirm the quality of meshing. In order to simplify the numerical simulation of the model the following assumptions were made:

- The walls is of coupled wall
- The fluid flow and the heat tranfer are in steady state and turbulent
- The shell side fluid is having constant thermal properties
- Any leakage of flow in the system is neglected
- the natural convection induced by the fluid density variation is neglected
- The temperature of the tube walls is kept constant
- The heat exchanger is well insulated so that the heat loss to the environment is totally neglected.

Computational and Solution Methods

Computational Fluid Dynamics, ANSYS Fluent 15.0 code was used to compute the flow properties within the flow path of the shell and tube domain using a steady time and absolute velocity formation and pressure-based solver. Acceleration due to gravity along x, y, z, axes were taken to be 0, -9.81m/s^2 and 0 respectively. The Energy equation was inserted in the model. For the viscous model, K-epsilon (2-equation), K-epsilon model (Realizable) and scalable wall functions were selected. The materials for the design of shell and tube model were also selected as;

For fluid – water as the liquid;
For solid- Aluminum, copper and steel were selected for the materials and the properties of these materials are as shown in Table 2.

Table 2: Properties of Materials considered

Materials	Thermal conductivity W/mK	Specific heat J/kg.K	Density kg/m^3
Copper	387.6	381	8978
Aluminum	202.4	871	2719
Steel	16.27	502.48	8030

The conditions of cold and hot stream of fluids were selected as:
Inlet cold: inlet velocity of 0.3m/s and temperature of 12^{o}C
Inlet hot: inlet velocity of 0.3m/s and temperature of 62^{o}C
Outlet Hot and cold: Backflow total temperature of 20^{o}C

And the wall conditions of the material used were also selected as:

- Heat transfer coefficient of 10 W/m^{2}k
- Free stream temperature 23^{o}C.

The reference value was computed from Inlet cold parameters: Area of $1m^2$, Density of $998kg/m^3$, Enthalpy of 1879J/kg, length of 1000 mm, temperature 12^oC, velocity 0.3m/s and the Ratio of Specific Heats is 1.4.

Under the *solution methods;*

SIMPLEC scheme– Pressure-Velocity Coupling with skewness correction of zero;

For the *Spatial Discretization*

Gradient – Green-Gauss Node Based; Pressure – Second order; Momentum – Second order Upwind; Turbulent Kinetic Energy - Second Order Upwind; Turbulent Dissipation Rate - Second Order Upwind; Energy - Second Order Upwind.

Under – Relaxation Factors

Pressure: 0.3; Density: 1; Body forces: 1; Momentum: 0.7; Turbulent kinetic energy: 0.8; Turbulent dissipation rate: 0.8; Turbulent viscosity: 1; Energy: 1.

Standard method was used and solution was initialized from the Inlet cold with temperature of 12^0C. Considering all the above boundary conditions and solution initialize condition, the number of iterations was set for 1200.The above solution was carried out for the three selected materials (Aluminum, Copper and steel).

The equation $Q = mC_p\Delta T$ was used to calculate the heat transfer rate across the shell side.
Where m is mass flow rate. C_p = specific Heat of water, ΔT= change in the temperature in the tube side.

3. RESULTS AND DISCUSSION

The result of modelled shell and tube heat exchanger is as shown in Figure 1. The result of meshing carried out on the modelled gives the computational domain of the heat exchangers shown in Figure 2. The mesh quality is as shown in Table 3 which show that the model is of good mesh quality since the standard deviation of different mesh quality is very low. The mesh metrics generated (Figure 3) show that 0.005 meshing gives 1293120 nodes and 4124027 elements. The solution converged at 223rd iteration for the three materials. The residual plot is shown in Figure 4. The x- velocity, y-velocity, z- velocity, k-epsilon was greater than 10^{-4} and the energy was greater than 10^{-7}. The pattern of cut plot and the charts are the same for the three materials (aluminum, copper and steel).

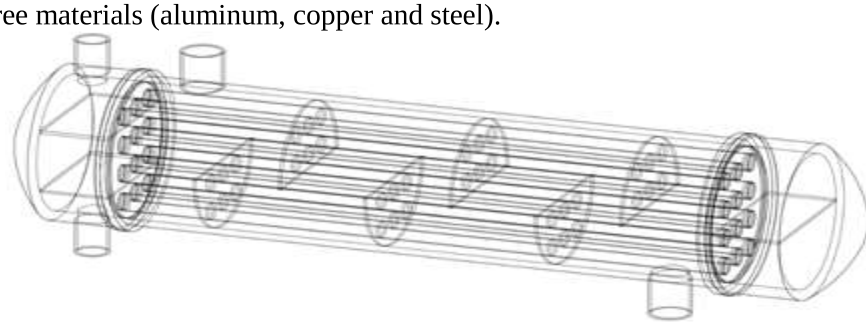

Figure 1: Modelled Shell and Tube Heat Exchanger Assembly

Table 3: The Mesh quality determined using the statistical measure.

Minimum	0.102203326359203
Maximum	0.999999949065753
Average	0.824650345057527
Standard Deviation	0.105906692144458

P J E A S
Vol. 1, No. 4, 2020 ISSN:2734-3170

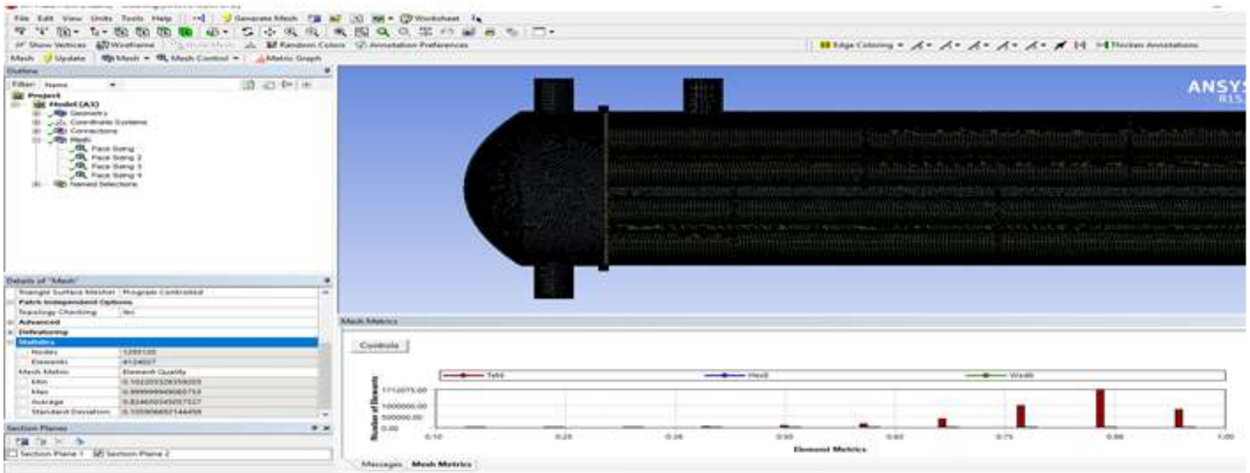

Figure 2: The Meshing Diagram of the Heat Exchanger

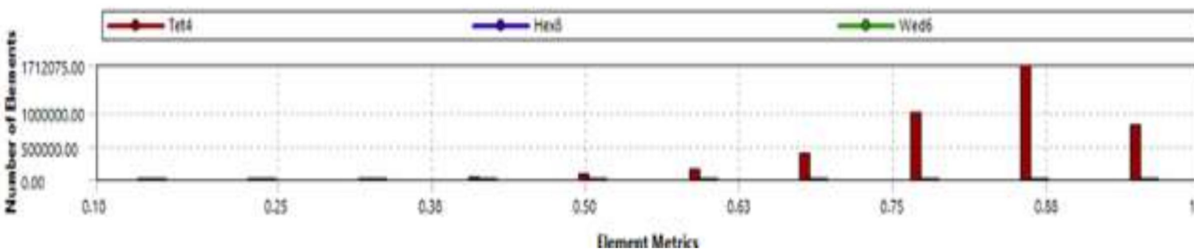

Figure 3: Mesh Metrics Generated

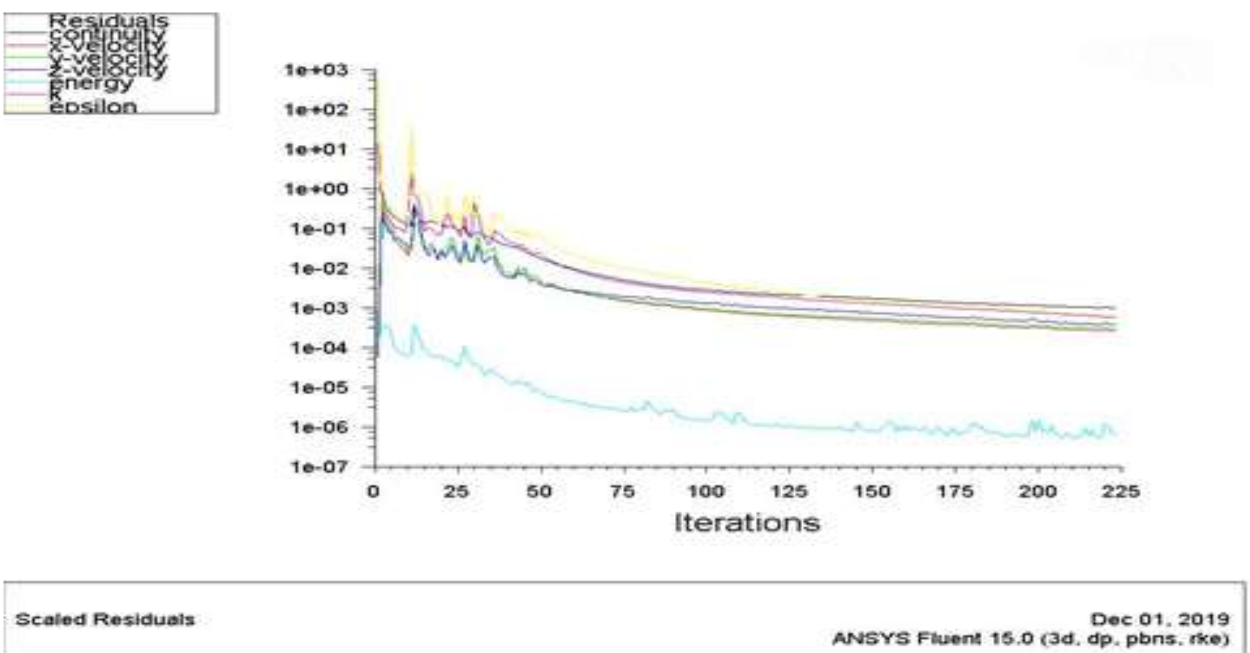

Figure 4: Scaled Residuals for Aluminum, Copper and Steel

Temperature distribution

Figure 5 illustrates the change in temperature of water in the hot and cold region of the heat exchanger made of each of the three selected materials. Temperature of the hot water in shell and tube heat exchanger at inlet and outlet were 335K and 329K respectively as shown in Table 4. In case of cold water, inlet temperature was 285K and the outlet became 291K. Temperature difference in the hot and cold are the same. Similar results were reported by Arjun and Gopu (2014).

The Inlet cold temperature equals 285K; The Outlet cold temperature equals 291 K

Temperature difference of 291 K – 285 K = 6 K

The Inlet hot temperature equals 335K; The Outlet cold temperature equals 329 K

Temperature difference =335 – 329 K = 6 K

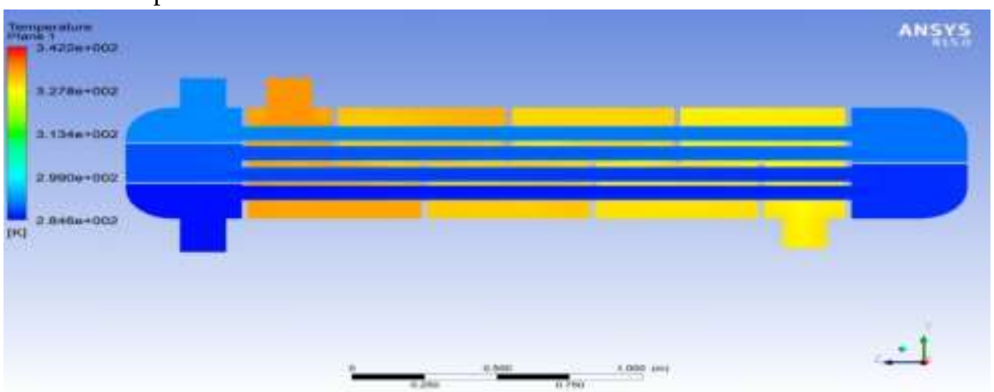

Figure 5: Temperature Distribution at different Points in the Shell and Tube

Table 4: Result of temperature variation obtained using the three materials

	Aluminum	**Copper**	**Steel**
Hot inlet	335 K	335 K	335 K
Cold inlet	285 K	285 K	285 K
Hot outlet	329 K	329 K	329 K
Cold outlet	291K	291 K	291 K

Pressure distribution

Figure 6 shows the contours pressure to give a detailed idea of pressure distribution across shell and tube heat exchanger. Table 5 presented the result of pressure variation obtained using the three materials. The table illustrated that the pressure difference in the hot region is 152 Pa – 0.610 Pa = 151.39 Pa for the three materials. While the cold region pressure difference is 938 Pa – 0.713 Pa = 937.287 Pa.

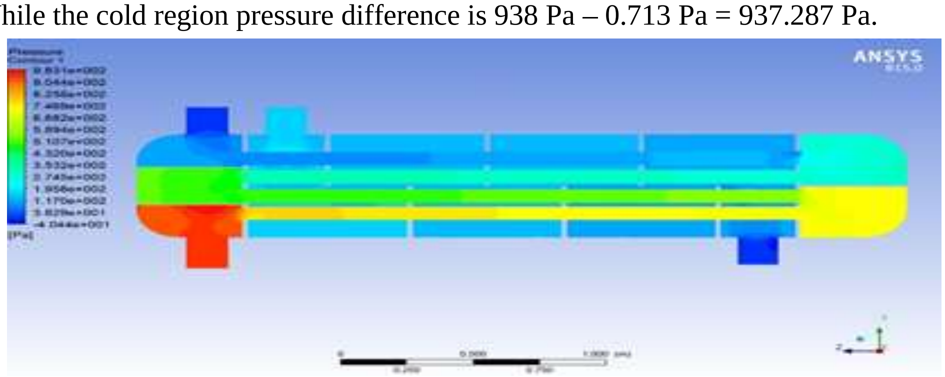

Figure 6: Pressure Distribution at different Points in the Shell and Tube

Table 5: Result of Pressure variation obtained using the three materials

	Aluminum	Copper	Steel
Hot inlet	152 Pa	152 Pa	152 Pa
Cold inlet	938 Pa	938 Pa	938 Pa
Hot outlet	0.610 Pa	0.610 Pa	0.610 Pa
Cold outlet	0.713 Pa	0.713 Pa	0.713Pa

Velocity distribution

The flow distribution across the cross section at different positions in heat exchanger is understood by the examination of velocity profile. Velocity profile of shell and tube heat exchanger at different points are shown in Figure 7 and Table 6. The velocity at the outlet is higher than the inlet velocity for cold and hot region for the three materials. Table 7 shows that the heat transfer coefficient for the hot and cold region is higher at outlet of the three materials and copper has the highest coefficient of heat transfer at hot outlet i.e 2059 W/(m^{2}K) .

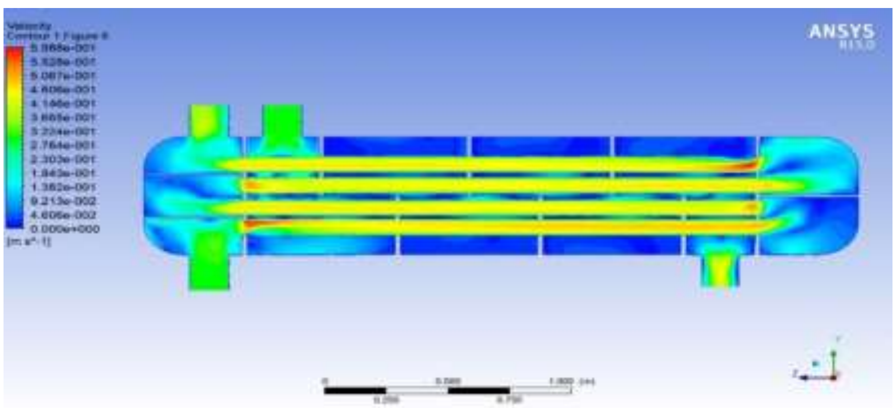

Figure 7: Velocity Distribution at different Points in the Shell and Tube

Table 6: Result of velocity variation obtained using the three materials

	Aluminum	**Copper**	**Steel**
Hot inlet	0.300 m/s	0.300 m/s	0.300 m/s
Cold inlet	0.300 m/s	0.300 m/s	0.300 m/s
Hot outlet	0.420 m/s	0.420 m/s	0.420 m/s
Cold outlet	0.368 m/s	0.368 m/s	0.368 m/s

Table 7: Result of heat transfer coefficient for the three materials

	Aluminum	**Copper**	**Steel**
Hot inlet W/m^{2}K	1478	1478	1478
Cold inlet W/m^{2}K	1532	1532	1532
Hot outlet W/m^{2}K	2049	2059	2045
Cold outlet W/m^{2}K	2037	2048	2041

4. CONCLUSION

ANSYS FLUENT 15.0 CFD package was used for the simulation of the characteristics of heat transfer of a shell and tube heat exchanger for different materials. The simulation was carried out for water to water heat transfer characteristics and for the same length and diameter of the shell and tube model, also for the same input temperature in the cold inlet 285K for hot inlet 335K for three materials (aluminum, copper and steel) using the k-epsilon. The pressure difference between hot and cold region vary widely i.e. 151.39 Pa and 937.287 Pa. In conclusion, according to the data gotten from the simulation in the post processing stage the temperature difference in aluminum is 6K, copper 6K and steel 6K, pressure outlet in cold region for materials is 0.713 Pa, pressure outlet in the hot region for the materials is 0.610 Pa, the velocity outlet in the cold region is 0.368 m/s and the velocity outlet in the hot region is 0.420 m/s. Copper has the highest overall heat transfer coefficient of 2059 W/(m^{2}K) amongst the three materials.

References

Arjun, K. S. and Gopu, K. B. (2014). Design of Shell and Tube Heat Exchanger Using Computational Fluid Dynamics Tools Research. *Journal of Engineering Sciences*, 3(7): 8-16.

Baru, D., Ibsa, N. and Venkatesa, P. (2018). CFD Simulation of a Double Pipe Heat Exchanger: Analysis Conduction and Convection Heat Transfer. *International Journal of Scientific Research and Review*, 7(12): 329-338.

Dawit, B. (2014). Design and Development of Shell and Tube Heat Exchanger for Harar Brewery Company Pasteurizer Application (Mechanical and Thermal Design). *American Journal of Engineering Research*, 3(10): 99-109.

Diaper, A. D. and Heseler, L. E. (1990). Crossflow Pressure Drop and Flow Distribution within a Tube Bundle Using Computational Fluid Dynamics. In Proceedings of 9th Heat Transfer Conference, Israel, pp. 235-240.

Dilip, S. P., Ravindrasinh, R. P. and Vipul, M. P. (2015). CFD Analysis of Shell and Tube Heat Exchangers –A review. *International Research Journal of Engineering and Technology*, 2(9): 2231-2235.

Haseler, L. E., Wadeker, V. V. and Clarke, R. H. (1992). Flow Distribution Effects in a Plate and Frame Heat Exchanger, IChemE Symposium Series, No. 129:361-367.

Huang, L. Y., Wen. J. X., Karayiannis, T. G. and Mathewws, R. D. (1996). CFD Modeling of Fluid Flow and Heat Transfer in a Shell and Tube Heat Exchanger. *The PHOENICS Journal of CFD and its Applications.* 9(2): 181-209.

Keene, L. W., Davies, T. W. and Gibbsons, D. B. (1994). Flow Distribution on the Shell-Side of a Shell-and-Tube Heat Exchanger, *Transaction of IchemE,* 71 (part A): 310-311.

Keene, L. W., Gibbsons, D. B., Evans, D. J. (1994). Computer Modeling Validation for Shell-and-Tube Heat Exchangers. *Transaction of IchemE,* 72, part A: 611-615.

Kolepaka, S. and Bicha, K. (2017). Comparative Analysis of Parallel and Counter Flow Heat Exchangers. *International Journal of Scientific Engineering and Technology Research*, 6(4): 638-644.

Lorenzini, E., Orlandelli, C. M., Spiga, M. and Tartarini, P. (1992). Experimental and Theoretical Investigation of a Tube-Bundle Heat Exchanger. *Institution Chemical Engineers Symposium Series*, 1(129): 459-465.

Pekdemir, T., Davies, T. W., Heseler, L. E., Diaper, A. D. (1994). Pressure Drop Measurements on the Shell-Side of a Cylindrical Shell-and-Tube Heat Exchanger. *Heat Transfer Engineering*, 15(3): 42-56.

Prekdemir, T., Devies, T. W., Haseler, L. E., Diaper, A. D. (1993). Flow Distribution on the Shell Side of a Cylindrical Shell-and-Tube Heat Exchanger. *International Journal of Heat and Fluid Flow*, 14(1): 76-85.

Rajiput, R. K. (2012). Heat and Mass Transfer. 5th edition, S. Chand nd Company PVT. Limited, New Delhi, India.

Revesz, Z. (1992). Convergence Behavior of Three-Dimensional Two-Medium Finite Difference computation to Model Shell-and-Tube Heat Exchangers. *Advance Computational Methods in Heat Transfer*, 2: 317-326.

Santosh, K. K. and Anandkumar S. M. (2017). CFD Analysis of Shell and Tube Heat Exchanger for Heat Transfer Capabilities. *International Journal of Engineering and Techniques*, 3(6):102-106.

Sneha. H. D., Kumar, E. M., Yeswanth, I. V. S. and Lakshmi, J. (2018). CFD Analysis on Concentric Tube Heat Exchanger in Parallel and Counter Flow Direction. *International Journal of Engineering Research and Application,* 8(6): 20-25.

Yang, S. R., Li, N., Zhao, H. and Zhang, C. J. (1990). On-Line fouling Monitoring Technique for Shell-Tube Heat Exchangers. In Proceedings of 9th Heat Transfer Conference, Israel, pp. 223-227.

Zhang, C. and Sousa, A. C. M. (1990). Numerical Simulation of Turbulent shear flow in an Isothermal Heat Exchanger Model. *Transactions of the ASME Journal of Fluid Engineering*, 112: 48-55.

Zhang, C., Sousa, A .C. M. and Venart, J. E. S. (1993). The Numerical and Experimental Study of a Power Plant Condenser. *Journal of Heat Transfer*, 115: 435-445.